ProtaStructure Design Guide

ProtaStructure™ and ETABS™ Result Comparison

Version 1.0 March 2025

Please contact us for your training and technical support queries

asiasupport@protasoftware.com

globalsupport@protasoftware.com

Publisher

PROTA SOFTWARE

Limitation of Responsibilities

Prota shall not be held responsible for any losses caused by documentation, software, or usage errors.

In addition to Prota License Agreement Terms, it is the responsibility of the user.

- to check results generated by documentation and software,
- make sure that the users of the software and their supervisors have adequate technical capabilities,
- Make sure the software is correctly used by reference manual and documentation.

Intellectual Property

ProtaStructure is a registered trademark of **Prota Software Inc,** and all intellectual property belongs to **Prota Software Inc**. Documentation, training, and reference manuals. Any program component cannot be copied, distributed, or used in violation of the license agreement.

Trademarks

ProtaStructure®, **ProtaDetails®**, **ProtaSteel®** ve **ProtaBIM®** are registered trademarks of Prota Software Inc. Prota logo is a trademark of Prota Software Inc.

Table of Contents

Model Description

In this case study, ETABS and ProtaStructure program results were compared using the ELF (Equivalent Lateral Force) method per **ASCE 7-16 code**.

Every structure and portion thereof, including nonstructural components that are permanently attached to structures and their supports and attachments, shall be designed and constructed to resist the effects of earthquake motions per Chapters 11, 12, 13, 15, 17 and 18 of ASCE 7-16.

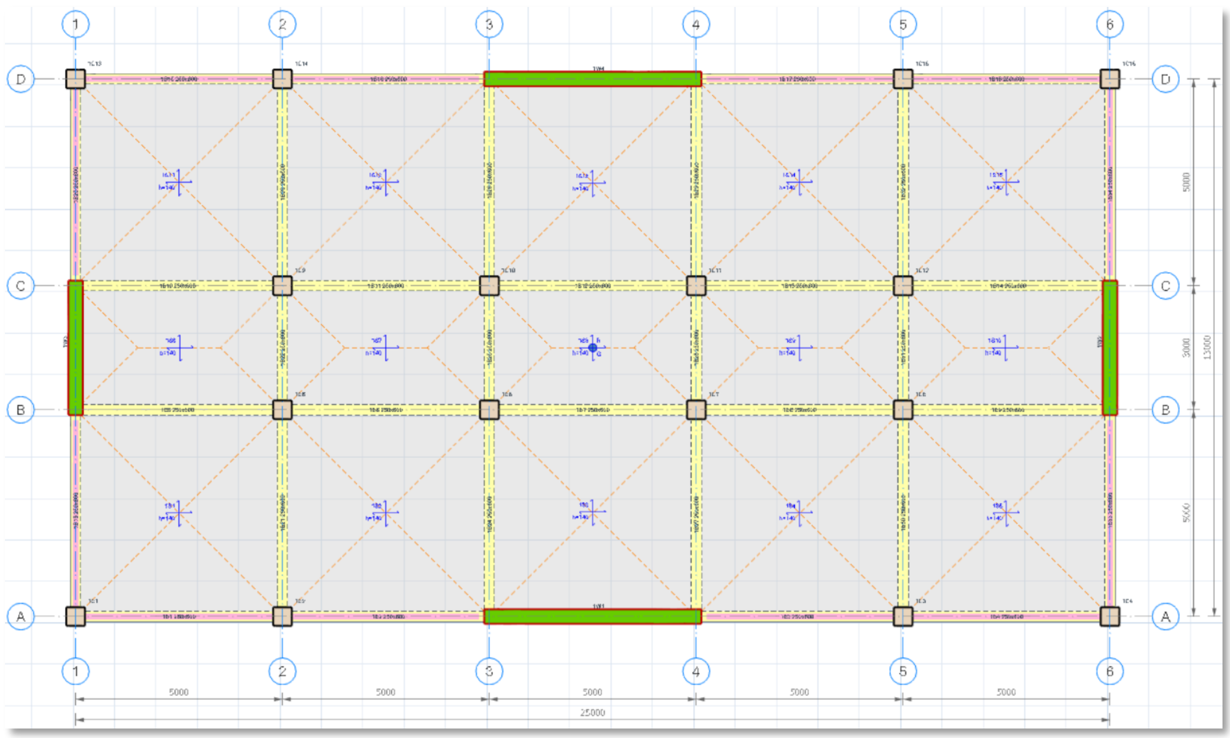
The case study building is composed of **15** stories. All the story heights are **3.5 m** except first story height which is **4 m**. As a result, the total structural height is **h_n=46 m**.

All stories are designed as reinforced concrete slabs with beams. There are shearwalls along both directions.

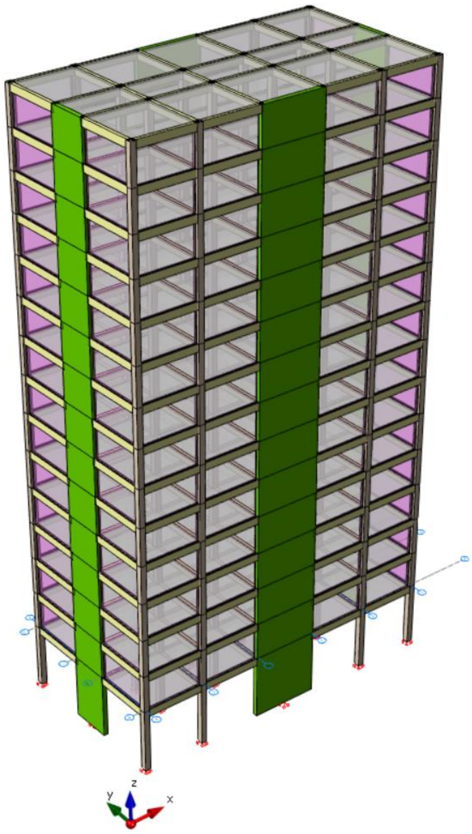
Concrete and reinforcement grades are selected as **C300** and **SD60** respectively. Slab and shearwall thicknesses are 140 mm. and 350 mm, respectively. Furthermore, all Beams are 250x600 mm. and all Columns are 450X450 mm.

Note:
The creation of the building model and the completion of the building analysis in ProtaStructure are out of scope of this study. For the sake of a more independent comparison, The ETABS Model was created from scratch in ETABS instead of exporting from ProtaStructure. Specific attention also has been paid to include the combined effect of gravity, lateral earthquake and vertical earthquake simultaneously (with cracked section properties) to exhibit a real-life scenario as much as possible, instead of simple comparison. Reasons for observed differences were explained wherever required.

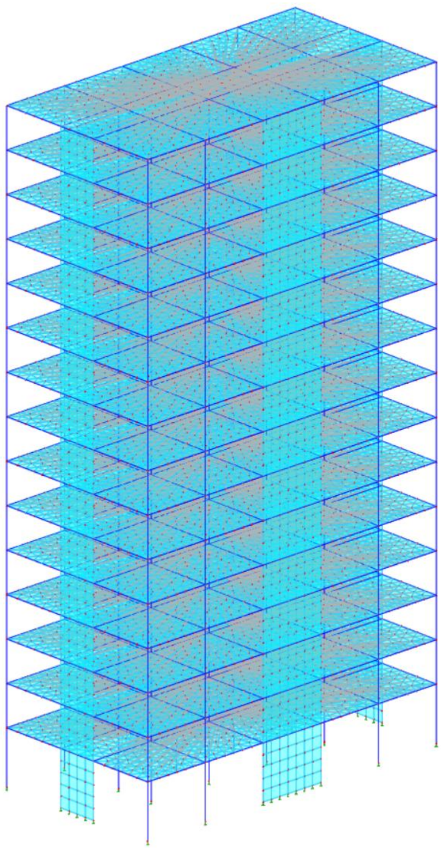
Plan and 3-D view of the building are shown below.



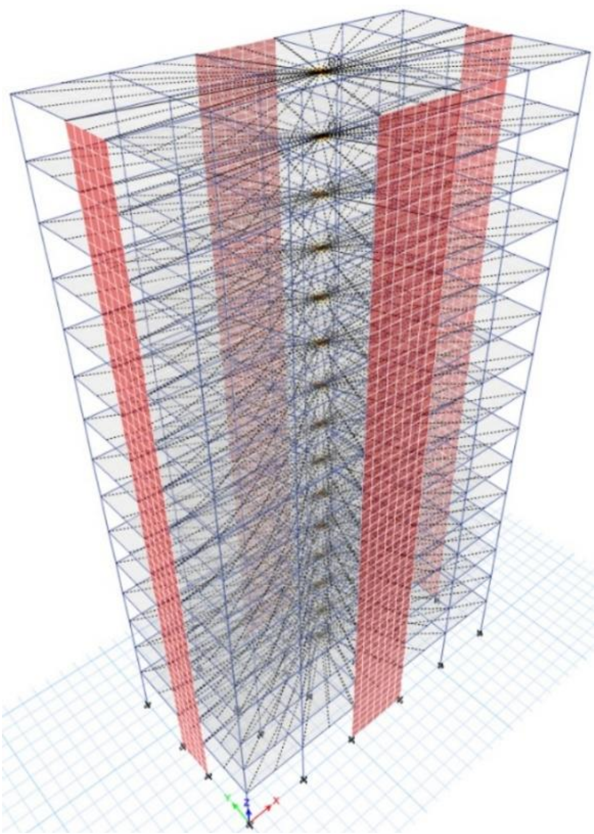
Typical Story Plan



ProtaStructure - 3D Physical Model



ProtaStructure - Analytical Model



ETABS - Analytical Model

Analytical Model

The SI unit system is used throughout the document. In both software, cracked section properties were used according to ACI 318-19 Code Section 6.6.3.1.1.

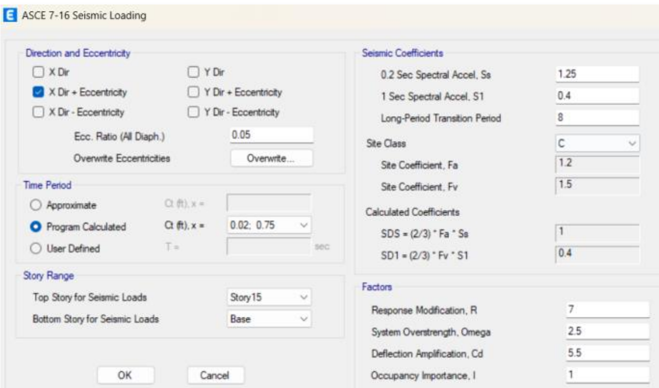
Remark:
ProtaStructure has a special feature that allows the use of cracked section properties for G and Q load cases which are part of seismic combinations and uncracked section properties for G and Q load cases that are part of non-seismic combinations. However, this requires two separate analyses, which ETABS cannot do automatically. ProtaStructure can consider both cracked and uncracked section properties in a single analysis run. This is often required by seismic codes to consider cracked and uncracked section scenarios simultaneously for seismic and non-seismic combinations.

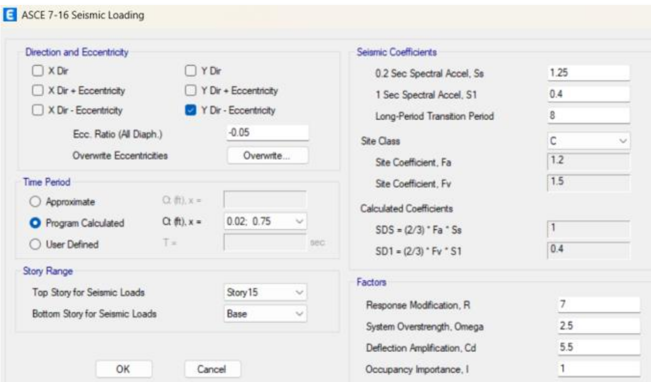
Note:
The existence of this feature does not affect the comparison since the seismic combinations are compared and the same cracked section properties are used in both software. We would need to make separate analyses in ETABS with uncracked section properties to compare the vertical combination results with ProtaStructure. Alternatively, we would need to set all cracked section modifiers to 1.0 in ProtaStructure.

Remark:
Rigid zones were not considered in modeling.

Load Pattern Definitions in ETABS

Load Pattern definitions in ETABS for **EXP** and **EYN** load cases are shown below.





Seismic Parameters

The mapped maximum considered earthquake spectral response accelerations, S_s **(for short period, 0.2 sec.)** is **1.25 g** and S_1 **(for long period, 1.0 sec.)** is **0.40 g**.

The building is located on **Site Class C** soils [ASCE 7-16 Table 20.3-1].

Short-period site coefficient, F_a **= 1.2** and long-period site coefficient, F_v **= 1.5** [ASCE 7-16 Tables 11.4-1 and 11.4-2].

Maximum considered earthquake spectral response accelerations adjusted for site class effects are determined by using the Equation 11.4-1 and Equation 11.4-1 in ASCE 7-16:

$S_{MS} = F_a\, S_s \quad = 1.20 \times 1.25 \quad = 1.500$ g

$S_{M1} = F_v\, S_1 \quad = 1.50 \times 0.4 \quad = 0.600$ g

The 5 % damped design spectral response accelerations $\mathbf{S_{DS}}$ at short period and $\mathbf{S_{D1}}$ at long period are determined by using the Equation 11.4-3 and Equation 11.4-4 in ASCE 7-16:

$S_{DS} = 2/3\, S_{MS} = 2/3 \times 1.500 \quad = 1.000$ g

$S_{D1} = 2/3\, S_{M1} = 2/3 \times 0.600 \quad = 0.400$ g

The Risk Category of the building is determined as **II** [Table 1.5-1 in ASCE 7-16].

Based on the Risk Category, the Seismic Importance Factor of the structure is I_e **= 1.00** [ASCE 7-16 Table 1.5-2].

Risk Category from Table 1.5-1	Snow Importance Factor, I_s	Ice Importance Factor—Thickness, I_i	Ice Importance Factor—Wind, I_w	Seismic Importance Factor, I_e
I	0.80	0.80	1.00	1.00
II	1.00	1.00	1.00	1.00
III	1.10	1.15	1.00	1.25
IV	1.20	1.25	1.00	1.50

All the structures shall be assigned to a **Seismic Design Category (SDC)** based on their risk category and the design spectral response acceleration parameters. Seismic Design Category (SDC) determines the permissible structural systems, limitations on height and irregularity, those components of the structure that must be designed for seismic loads, and the types of analysis required.

The determination of **SDC** is carried out by using Table 11.6-1 and Table 11.6-2 in ASCE 7-16.

$\mathbf{S_{DS}}$ = 1.000 g and Risk Category is II $\Rightarrow$ SDC is D.

$\mathbf{S_{D1}}$ = 0.400 g and Risk Category is II $\Rightarrow$ SDC is D.

Therefore, **Seismic Design Category** is determined as **D**.

According to ASCE 7-16 Table 12.2-1.

D. DUAL SYSTEMS WITH SPECIAL MOMENT FRAMES CAPABLE OF RESISTING AT LEAST 25% OF PRESCRIBED SEISMIC FORCES	12.2.5.1							
1. Steel eccentrically braced frames	14.1	8	2½	4	NL	NL	NL	NL
2. Steel special concentrically braced frames	14.1	7	2½	5½	NL	NL	NL	NL
3. Special reinforced concrete shear walls[g,h]	14.2	7	2½	5½	NL	NL	NL	NL
4. Ordinary reinforced concrete shear walls[g]	14.2	6	2½	5	NL	NL	NP	NP
5. Steel and concrete composite eccentrically braced frames	14.3	8	2½	4	NL	NL	NL	NL
6. Steel and concrete composite special concentrically braced frames	14.3	6	2½	5	NL	NL	NL	NL

The "**Seismic Force Resisting System**" is selected as "**D3**" and the corresponding "**Response Modification Coefficient**" is **R=7**.

Overstrength Factor, Ω_0 is **2.50** and **Deflection Amplification Factor, C_d** is **5.50**. It can be seen from the table that the structural height is not limited for SDC=D.

Selection of Analysis Procedure

After carrying out the building analysis process in ProtaStructure, it's necessary to check whether the ELF (Equivalent Lateral Force) method is suitable for the structure under consideration.

For this purpose, horizontal and vertical irregularities can be checked using "**Post-Analysis Checks Report**" under the Reports tab in ProtaStructure.

In the Notifications tab, it's indicated that there is no horizontal and vertical irregularity for the structure.

- Dir 1: Stiffness Irregularity does not exist.
- Dir 2: Stiffness Irregularity does not exist.
- Interstorey Mass Irregularity does not exist.
- Interstorey Strength Irregularity (Weak Storey) does NOT exist in the building.
- Dir 1: Torsion Irregularity does not Exist.
- Dir 2: Torsion Irregularity does not Exist.
- Dir 1... Relative Storey Drifts satisfies the Limits.
- Dir 2... Relative Storey Drifts satisfies the Limits.

Considering that total structural height is **h_n=46 m**. and **SDC is D**, ELF method can be used according to ASCE 7-16 Table 12.6-1 as shown below.

Table 12.6-1 Permitted Analytical Procedures

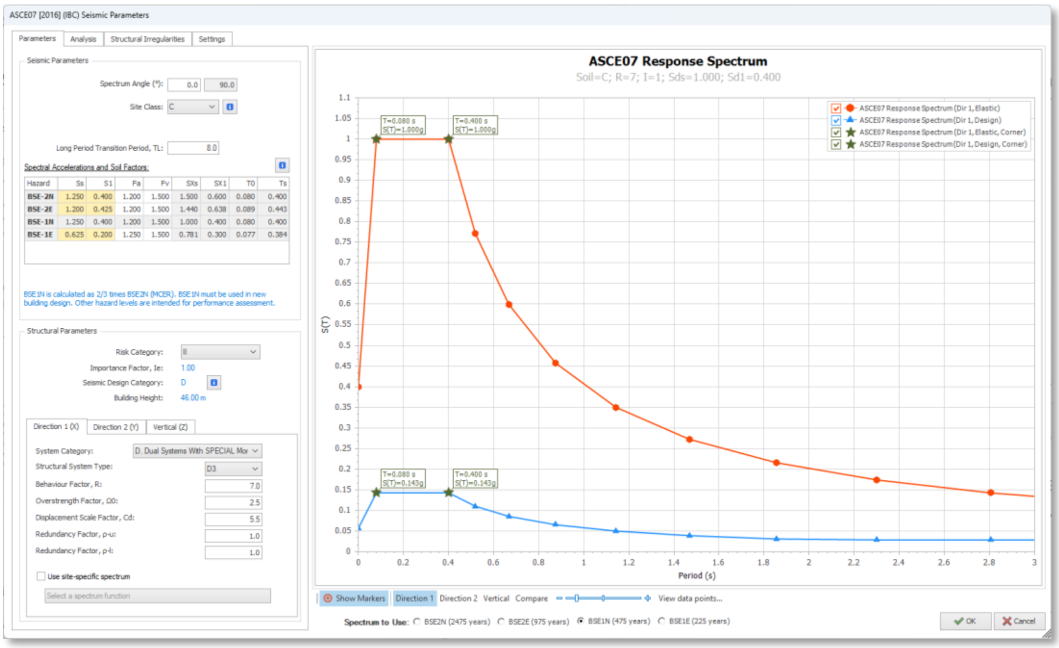
Seismic Design Category	Structural Characteristics	Equivalent Lateral Force Procedure, Section 12.8[a]	Modal Response Spectrum Analysis, Section 12.9.1, or Linear Response History Analysis, Section 12.9.2[a]	Nonlinear Response History Procedures, Chapter 16[a]
B, C	All structures	P	P	P
D, E, F	Risk Category I or II buildings not exceeding two stories above the base	P	P	P
	Structures of light-frame construction	P	P	P
	Structures with no structural irregularities and not exceeding 160 ft (48.8 m) in structural height	P	P	P
	Structures exceeding 160 ft (48.8 m) in structural height with no structural irregularities and with $T < 3.5T_s$	P	P	P
	Structures not exceeding 160 ft (48.8 m) in structural height and having only horizontal irregularities of Type 2, 3, 4, or 5 in Table 12.3-1 or vertical irregularities of Type 4, 5a, or 5b in Table 12.3-2	P	P	P
	All other structures	NP	P	P

[a]P: Permitted; NP: Not Permitted; $T_s = S_{D1}/S_{DS}$.

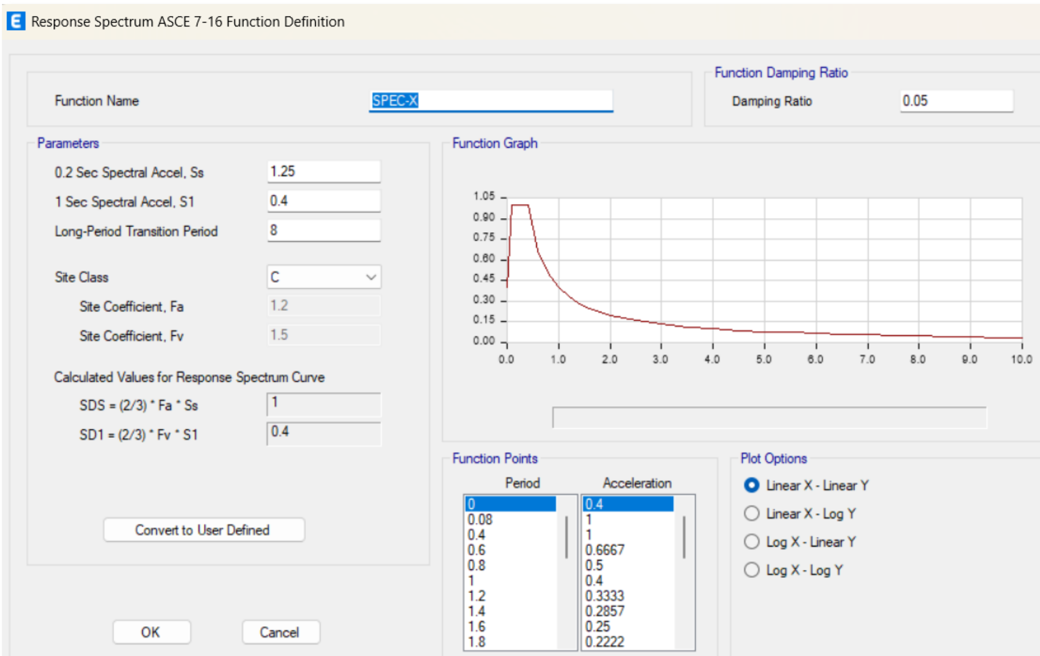
Horizontal Acceleration Spectra in ProtaStructure and ETABS

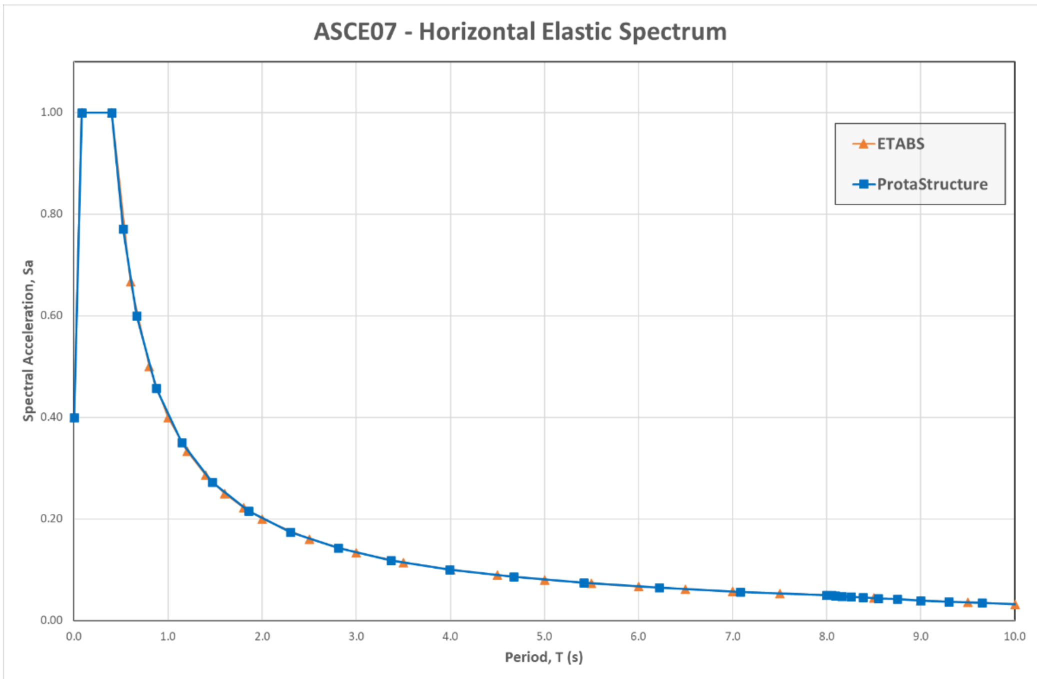
ProtaStructure calculates Design Response Spectrum for horizontal directions per ASCE 7-16 code as shown below.

It's the same for Direction 1 and Direction 2 for this model but can be selected differently for different earthquake directions.



The horizontal elastic spectrum in **ETABS** is as follows.





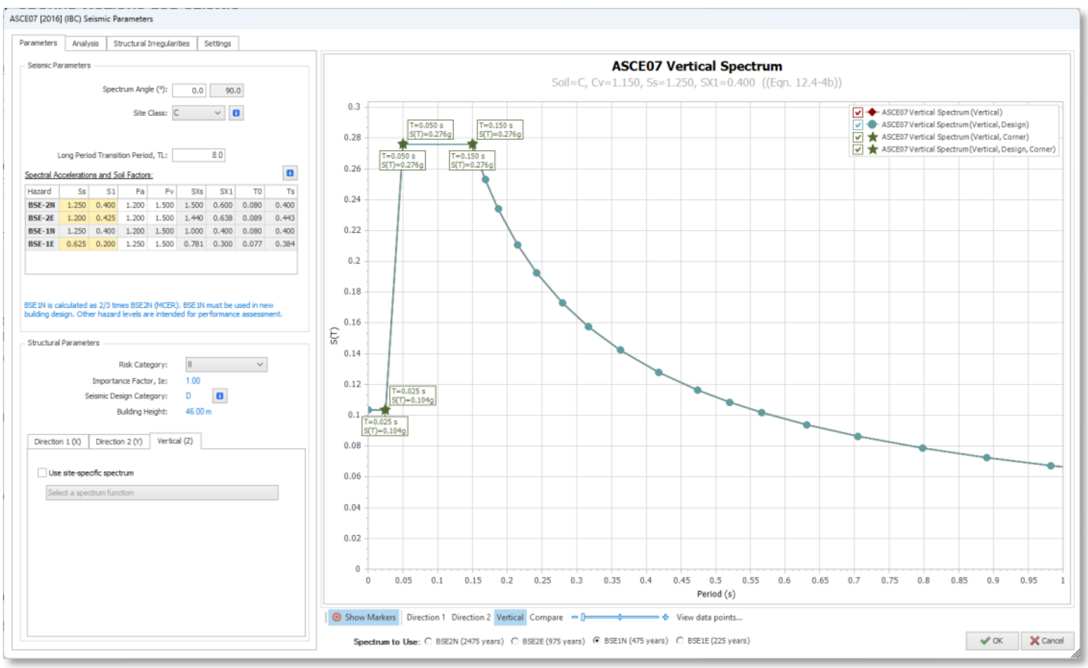
Horizontal elastic spectra in ProtaStructure and ETABS plotted on the same graph.

Vertical EQ Spectra in ProtaStructure and ETABS

For the structures in Seismic Design Categories C, D, E and F, vertical seismic load effect is determined with the following equation [ASCE 7.16 Equation 12.8-2]:

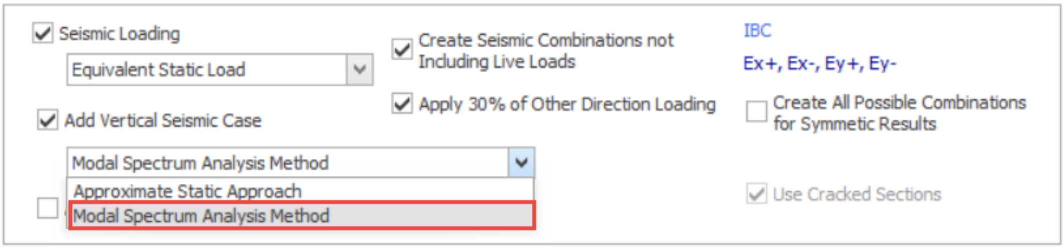
$$E_v = 0.3 S_{av} D$$

ProtaStructure calculates the vertical response spectrum by using the formulas given in ASCE 7.16 Section 11.9 and then multiplies S_{av} values by 0.3. It's shown below.



In ProtaStructure Vertical and Horizontal spectra are also tabulated and graphically presented in "**Pre-Analysis Checks Report**".

Vertical earthquake analysis is then performed with "**Modal Spectrum Analysis Method**".



For the structures in Seismic Design Categories A, B, vertical seismic load effect shall be determined by the "**Approximate Static Approach**".

Remark:
ETABS cannot automatically calculate vertical design spectrum per ASCE 7.16 code. Therefore, the vertical spectrum calculated by ProtaStructure was directly transferred to ETABS via "From File" option.

Comparison of ETABS and ProtaStructure Results

Storey Masses and Weights

A comparison of storey mass and seismic weights are tabulated below:

Storey	H	Mass (tons)		G (kN)		Q (kN)		W (kN)	
		ETABS	PS	ETABS	PS	ETABS	PS	ETABS	PS
15	3.0	264.78	281.20		2649.53		650.00	2941.25	2812.03
14	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
13	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
12	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
11	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
10	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
9	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
8	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
7	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
6	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
5	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
4	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
3	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
2	3.0	352.79	333.04		3167.93		650.00	3459.65	3330.43
1	4.0	364.50	356.02		3397.68		650.00	3689.40	3560.18
Total				49168.60	47230.32	9750.00	9750.00	51606.10	49667.82

Note:
Mass and seismic weight values align well between the two software. The **4.1%** difference is mainly due to the different approaches taken in structural self-weight calculation load decomposition between ETABS and ProtaStructure. No such difference is observed in imposed loads.

- ProtaStructure accurately calculates and decomposes slab self-weights faces of the beams. Therefore, the slab net span is used in calculations to prevent double counting with beam self-weights and the portion of any additional dead loads on the beam surface.

- Beam weights are not considered in the beam-column intersections in ProtaStructure, to avoid double counting which results in lower overall weight.

- ETABS distributes half of the first storey mass to the foundation level which is not given on the table above.

- ETABS does not report mass values of G and Q separately for each storey. That’s why they weren’t included in this table.

Modes and Modal Participating Mass Ratios

Results obtained in both programs are as follows

Mode	Periods (s)		Mass Participation along X-Axis (%)		Mass Participation along Y-Axis (%)		Mass Participation About Z-Axis (%)	
	ETABS	PS	ETABS	PS	ETABS	PS	ETABS	PS
1	2.207	2.235	0.000	0.000	73.500	74.130	0.000	0.000
2	1.647	1.670	69.780	70.091	0.000	0.000	0.000	0.000
3	1.515	1.539	0.000	0.000	0.000	0.000	70.690	71.201
4	0.614	0.634	0.000	0.000	13.320	12.838	0.000	0.000
5	0.401	0.412	16.200	15.845	0.000	0.000	0.000	0.000
6	0.384	0.397	0.000	0.000	0.000	0.000	15.190	14.276
7	0.285	0.300	0.000	0.000	5.540	5.396	0.000	0.000
8	0.172	0.175	0.000	6.455	0.000	0.000	0.000	0.000
9	0.171	0.174	6.150	0.000	0.000	3.016	0.000	0.000
10	0.166	0.115	0.000	0.000	0.000	1.824	6.150	0.000
11	0.166	0.099	0.000	3.357	2.960	0.000	0.000	0.000
12	0.164	0.082	0.290	0.000	0.000	1.137	0.000	0.000

There is a **1.25%** difference for the 1st mode period (Y-direction mode) and a **1.38%** difference for the 2nd mode period (X-direction mode) between ETABS and ProtaStructure.

As for the 3rd mode period (primarily rotation mode) there is a **1.56%** difference.

Note:
Modal results align well between two software within reasonable tolerances. It must be acknowledged that the slight discrepancy here results from the difference in mass calculation and calculation of mass moments of inertia (in addition to stiffness differences in FE formulation and assumptions)

Base Shear and Floor Shear Forces

Base Shear and lateral loads calculated at the floor levels obtained in both programs are tabulated as follows.

Storey	Fx & Fy (kN)		
	ETABS	PS	Difference
15	268.73	292.45	8.11%
14	326.83	315.93	3.45%
13	296.36	286.42	3.47%
12	266.70	257.69	3.50%
11	237.87	229.78	3.52%
10	209.93	202.73	3.55%
9	182.92	176.60	3.58%
8	156.92	151.45	3.61%
7	132.00	127.35	3.65%
6	108.26	104.39	3.71%
5	85.80	82.69	3.76%
4	64.79	62.40	3.83%
3	45.43	43.72	3.91%
2	28.04	26.96	4.01%
1	13.59	13.49	0.74%
Total	**2424.17**	**2374.06**	**2.11%**

Note:
The base shear results align well between two software within reasonable tolerances. Total base shear differs by **2.11%** between the two software. This is based on the difference between seismic weights and vibration periods, as well as differences in FE modeling approaches.

Manual Verification of Base Shear

ProtaStructure base shear value is verified against manual hand calculations in this section.

Fundamental Period of the Structure in X Direction **T (X) = 1.670 sec.**

Fundamental Period of the Structure in Y Direction **T (Y) = 2.235 sec.**

T_a = C_t H_n =0.0488 x 46 $^{0.75}$ = **0.862 sec.** [ASCE 7.16 Table 12.8-2]

T_{max} = C_u T_a = 1.4 x 0.862 = **1.207 sec.** [ASCE 7.16 Table 12.8-1]

T(X) = 1.670 sec. > T_{max} = 1.207 sec. then **T (X) = 1.207 sec.**

T(Y) = 2.235 sec. > T_{max} = 1.207 sec. then **T (Y) = 1.207 sec.**

In both X and Y directions, the upper bound value for the periods governs.

C_s = 1.000 / (7/1.00) = 1.0 / 7.0 = **0.1429** [ASCE 7.16 Equation 12.8-2]

C_s maximum (X Direction) = 0.4 / [(1.207) x (7.0/1.00)] = **0.0473**

C_s maximum (Y Direction) = 0.4 / [1.207 x (7.0/1.00)] = **0.0473**

C_s minimum (X and Y Direction) = 0.044 x 1.000 x 1.00 = **0.044** ≥ **0.01** √

∴ **Cs (X) = 0.0473**, **Cs(Y) = 0.0473**

W= G+nQ = 47230.32 + 0.25 x 9750 = **49667.82 kN**

V_X = Cs(X) W = 0.0473 x 49667.82 = **2349.29 kN**

V_Y = Cs(Y) W = 0.0473 x 49667.82 = **2349.29 kN**

V_{min}= Cs_{min} * S_{DS} * I_e = 0.044 x 49667.82 x 1.0 = **2185.38 kN**

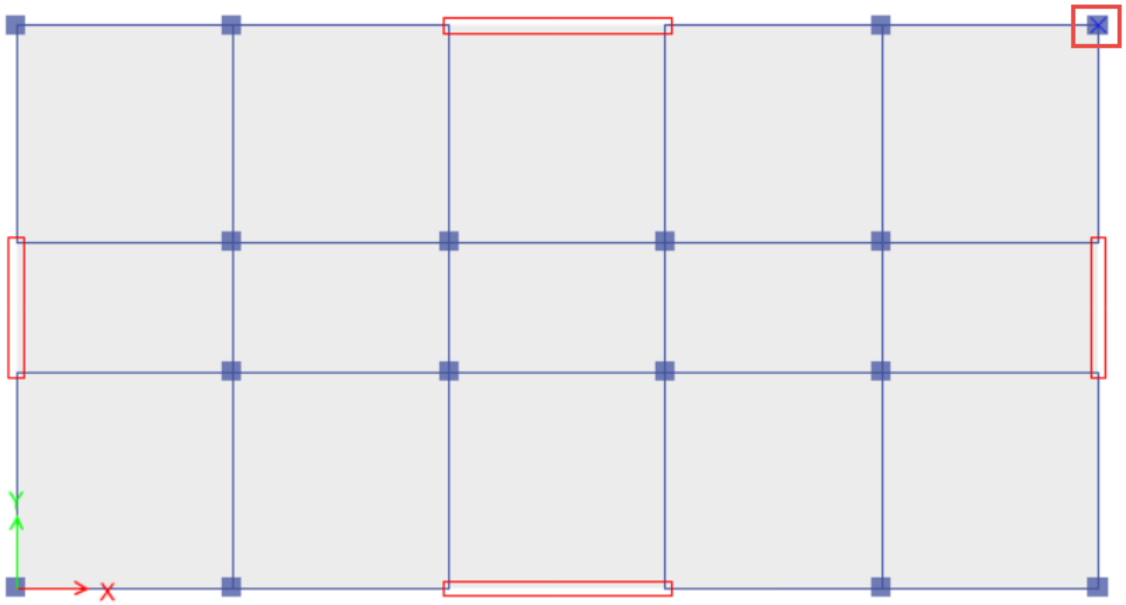
ProtaStructure results are summarized below.

Dir.	Period (s)		Limit Period (s)		Spectral Acceleration (g)		Total Base Shear (kN)		Minimum Base Shear (kN)	
	Hand Calc.	PS	Hand Calc.	PS	Hand Calc.	PS	Hand Calc.	PS	Hand Calc.	PS
1	-	1.670	1.207	1.212	0.0473	0.0480	2349.29	2374.06	2185.38	2191.59
2	-	2.235	1.207	1.212	0.0473	0.0480	2349.29	2374.06	2185.38	2191.59

Note:
ProtaStructure base shear results seem in line with hand calculations.

Comparison of Joint Displacements and Rotations

The top-rightmost joint of the 15th floor was considered for comparison.



Displacement and rotation values for **EYP** load case for this point in both programs are given below.

EYP Load Case	Ux (mm)	Uy (mm)	Uz (mm)	Rx (rad)	Ry (rad)	Rz (rad)
ETABS	-4.913	121.470	-3.643	-0.001284	-0.000009	0.000756
ProtaStructure	-5.755	127.361	-3.688	-0.001270	-0.000020	0.000890

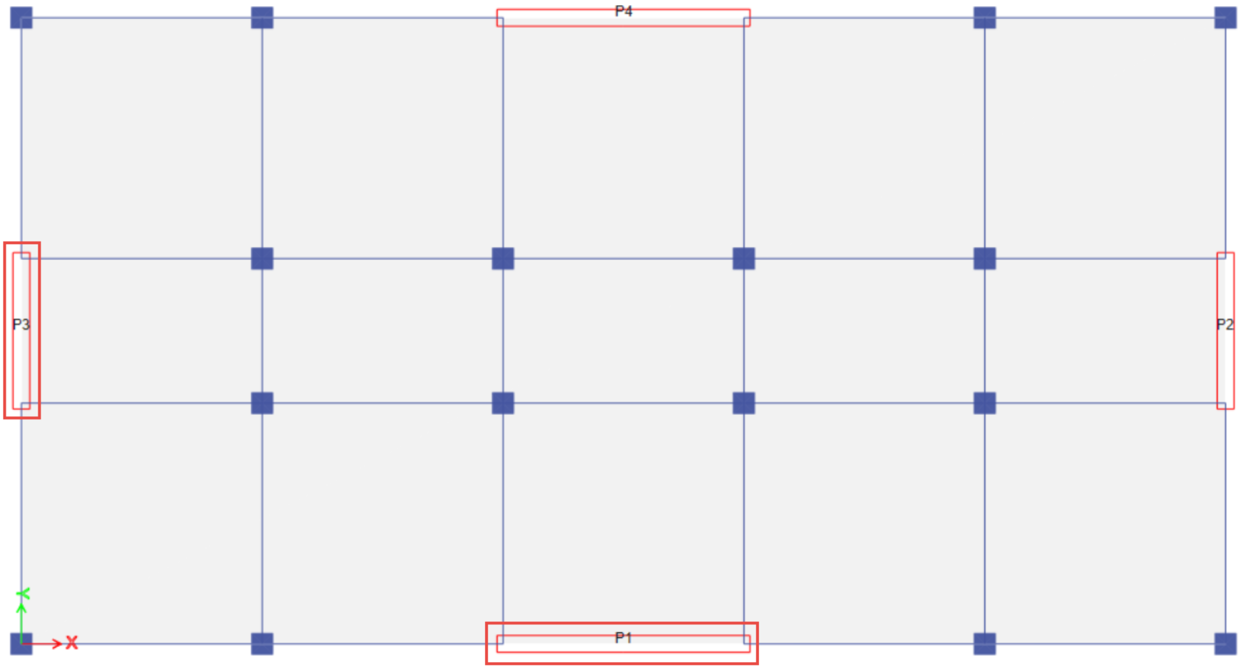
Displacement and rotation values for **EXP** load case in both programs are given below.

EXP Load Case	Ux (mm)	Uy (mm)	Uz (mm)	Rx (rad)	Ry (rad)	Rz (rad)
ETABS	69.139	-4.913	-1.323	0.000003	0.001084	-0.000393
ProtaStructure	72.653	-5.753	-1.370	0.000010	0.001120	-0.000460

Note:
The difference in results between two software seems to be within reasonable limits. The difference is due to the difference between seismic weights and vibration periods, as well as differences in FE modeling approaches and mesh size used in the models.

Comparison of Shearwall Internal Forces

The shearwalls **P1** and **P3** at **Storey 1** were considered for the comparison:

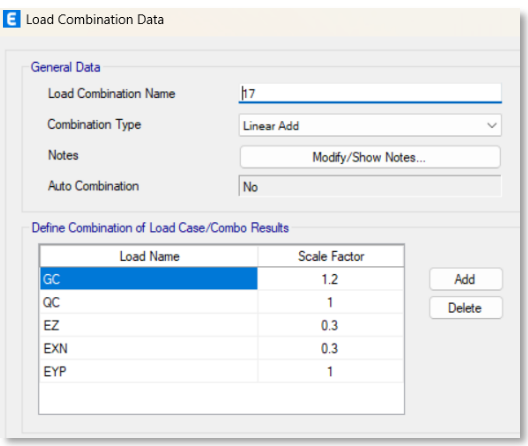


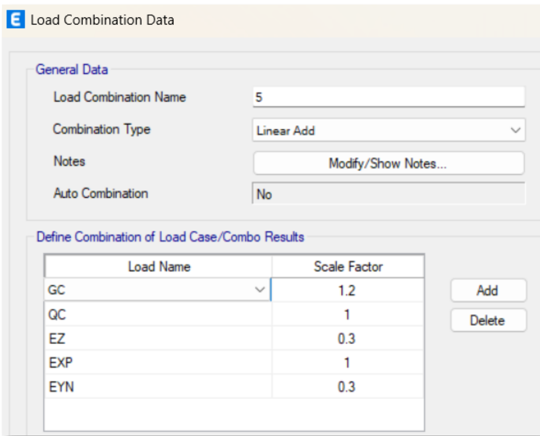
Load Cases and Combinations to Compare

To capture the results in major and minor axes of the shearwalls, results of two separate combinations and their relevant load cases were compared.

- 1.2G + Q + Ex + 0.3Ey + 0.3Ez (Combination 5, Dominant seismic load in X direction)
- 1.2G + Q + Ey + 0.3Ex + 0.3Ez (Combination 17, Dominant seismic load in Y direction)

Combination information is shown in the following screenshot from ETABS





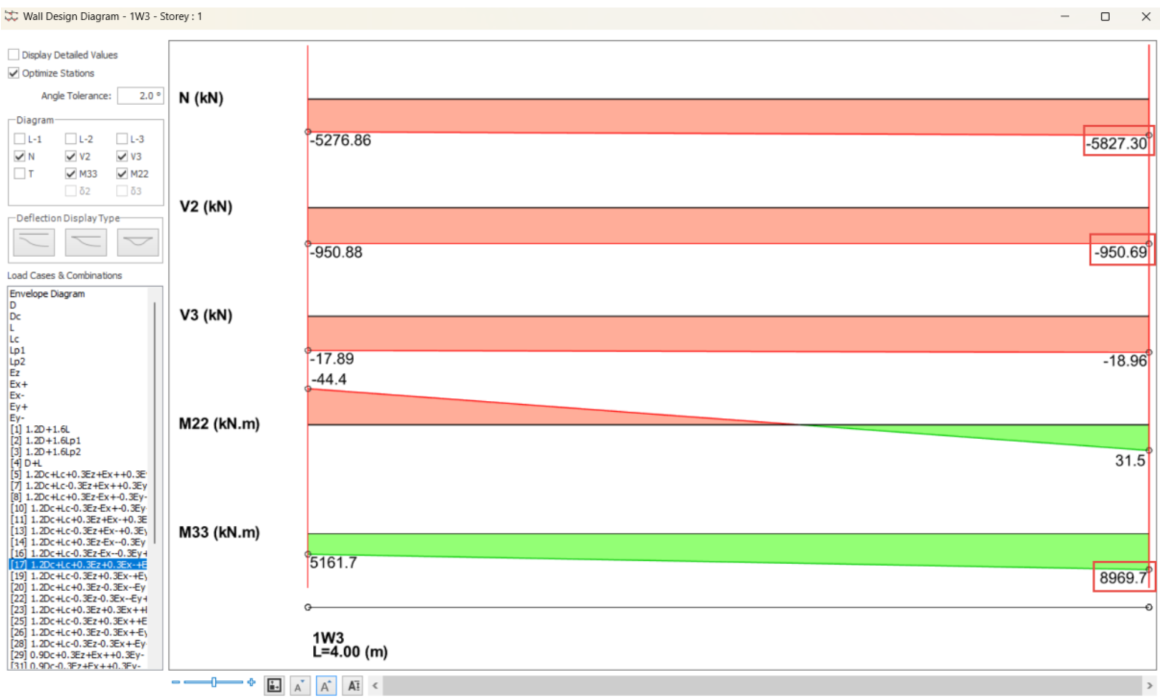
Results for shearwall P1 are given below

Shearwall P1	Axial Force (kN)		Shear (kN)		Moment (kNm)	
	ETABS	PS	ETABS	PS	ETABS	PS
G	-5883.29	-5653.77	0.00	0.03	0.00	-2.10
Q	-980.59	-966.25	0.00	0.01	0.00	-0.80
EXP	0.00	-0.15	1102.84	-1068.72	17798.32	16553.80
EYN	1390.18	1392.16	84.08	95.55	1435.04	-1533.10
EZ	922.82	-895.92	0.00	0.65	0.00	21.70
Comb. 5 1.2G+Q+EX+0.3EY+0.3EZ	-7346.64	-7601.58	1128.06	-1039.82	18228.84	16097.10

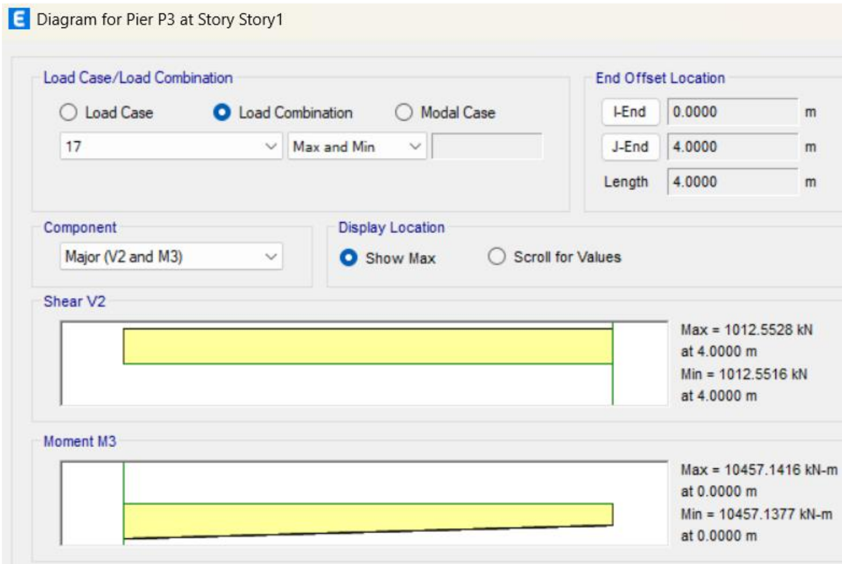
Results for shearwall P3 are given below

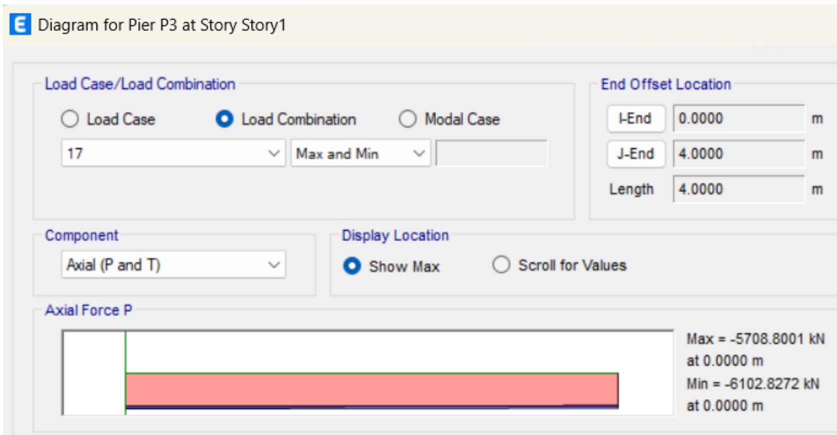
Shearwall P3	Axial Force (kN)		Shear (kN)		Moment (kNm)	
	ETABS	PS	ETABS	PS	ETABS	PS
G	-4482.48	-4258.77	0.00	0.01	0.00	-0.20
Q	-745.51	-727.20	0.00	0.00	0.00	-0.10
EXN	728.89	724.59	31.00	32.58	380.44	-371.70
EYP	0.00	0.02	1003.25	-960.57	10343.00	9079.20
EZ	656.71	-689.89	0.02	0.32	0.06	7.80
Comb. 17 1.2G+Q+EY+0.3EX+0.3EZ	-5708.80	-5827.30	1012.55	-950.69	10457.14	8969.70

ProtaStructure diagrams for Combination 17 are given below for reference:

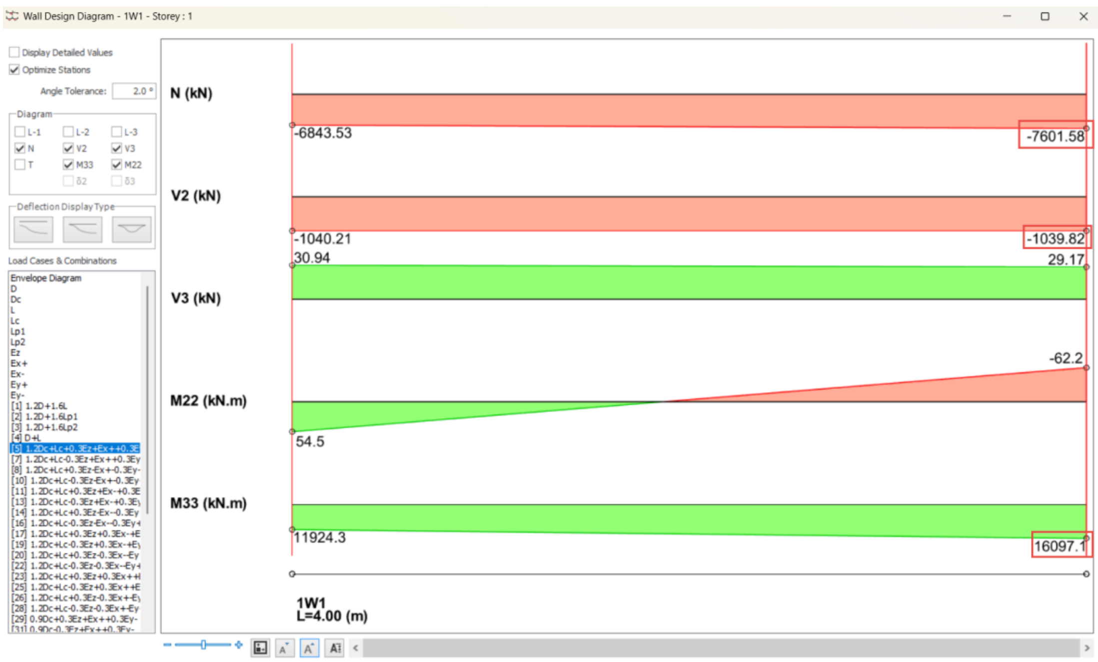


ETABS diagrams for Combination 17 are given below for reference:

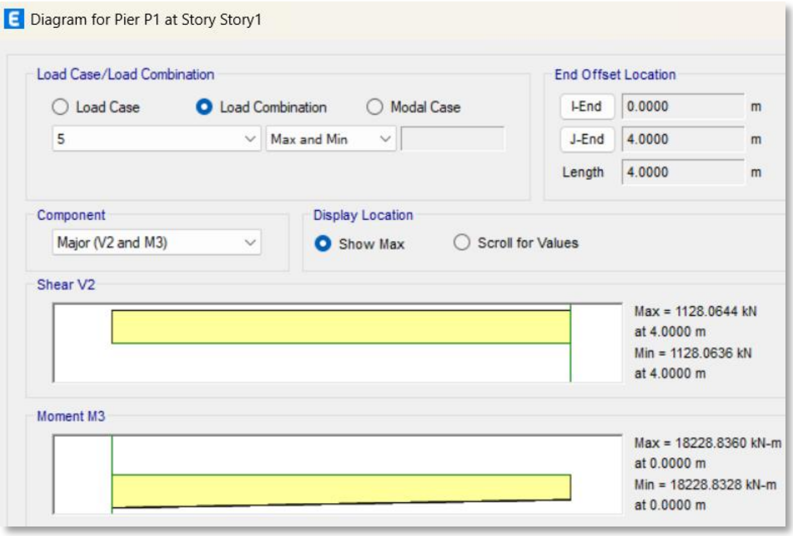


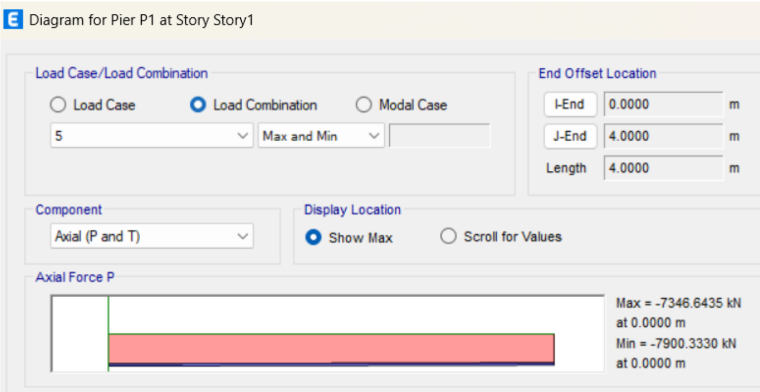


ProtaStructure diagrams for Combination 5 are given below for reference:



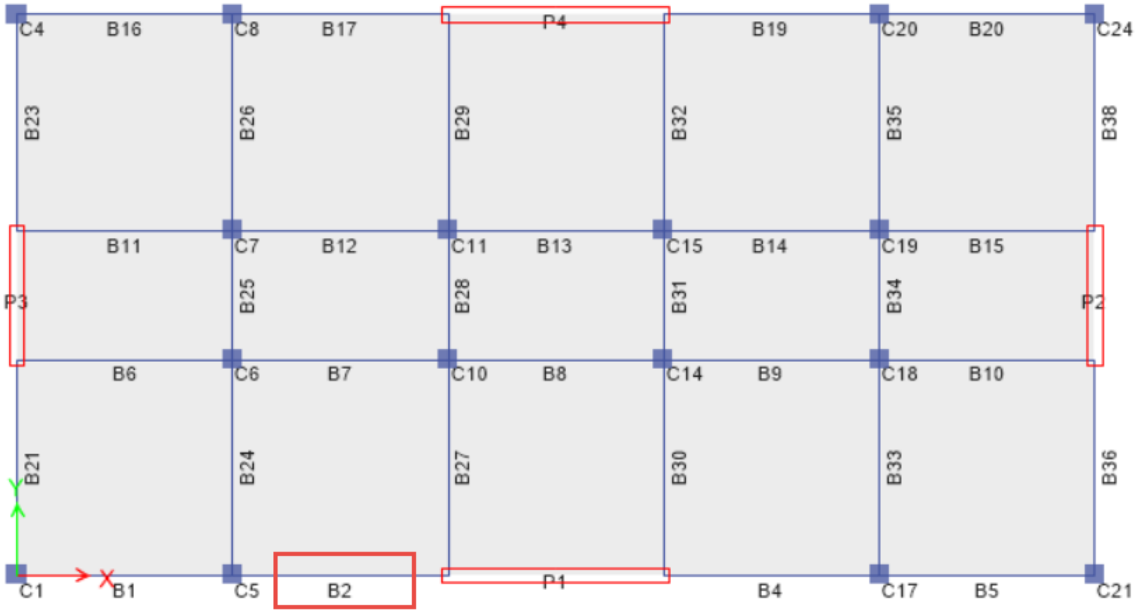
ETABS diagrams for Combination 5 are given below for reference:





Comparison of Beam Internal Forces

The beam **B2** at **Storey 1** was considered for the second comparison:



Combination-5 results for the beam B2 at Storey 1 are as follows.

Beam B2 – Storey 1	Shear Force – Left (kN)		Shear Force – Right (kN)	
	ETABS	PS	ETABS	PS
G	-40.16	-39.25	44.20	46.16
Q	-5.70	-5.76	6.66	7.15
EXP	14.81	15.02	13.04	15.55
EYN	0.89	-1.82	-0.32	-3.66
EZ	2.99	4.83	0.08	-5.37
Comb. 5 1.2G+Q+EX+0.3EY+0.3EZ	-37.90	-37.05	73.58	75.44

Beam B2 – Storey 1	Moment-Left (kN)		Moment - Span (kN)		Moment-Right (kNm)	
	ETABS	PS	ETABS	PS	ETABS	PS
G	-33.33	-34.09	18.11	18.57	-43.72	-48.9
Q	-5.34	-5.55	3.43	3.58	-7.64	-8.60
EXP	33.60	34.67	7.42	7.99	-35.94	-39.10
EYN	1.30	-4.96	-0.13	-1.57	-0.45	7.50
EZ	3.77	-5.57	1.84	-2.78	3.94	6.40
Comb. 5 1.2G+Q+EX+0.3EY+0.3EZ	-12.47	-15.00	34.64	32.70	-97.25	-102.20

Note:
The results align well between the two software and differences are within reasonable limits.

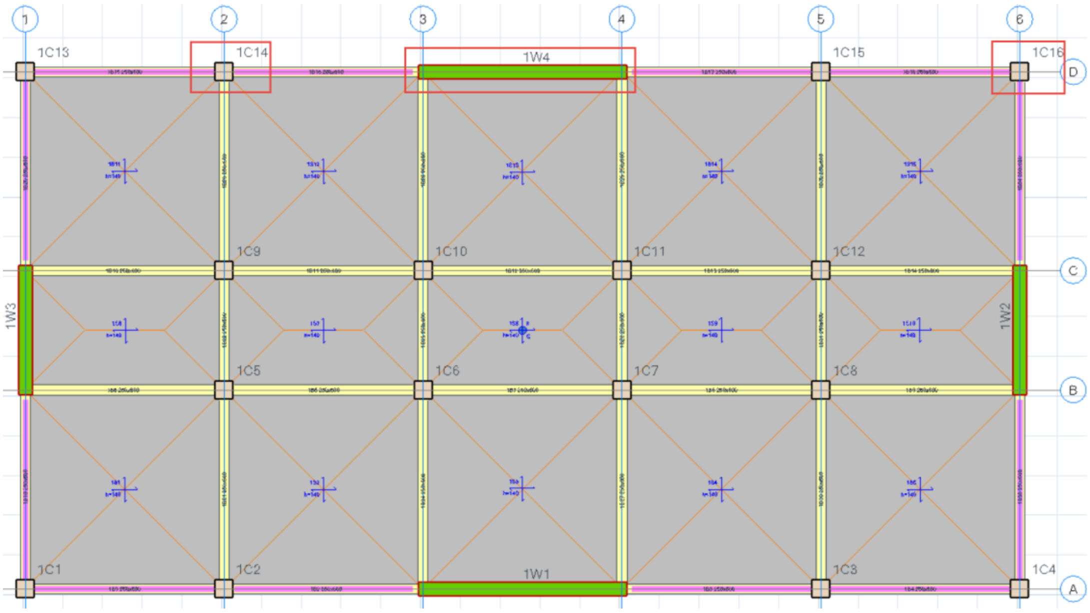
ProtaStructure utilizes "**relatively rigid**" **top chords** on the shearwall members to avoid unrealistic diminishing of beam support moments due to fine mesh sensitivity because of the edge-length sensitivity of drilling degrees of freedom in membrane element formulations. In addition to other differences in mass, weight, base shear and FE modeling approaches, this may be another reason for the slightly larger beam support moment.

Another contributing factor is the mesh size used in slabs, which affects the load and mass distribution on the beams. Especially, the mass distribution plays an important role in vertical earthquake effects on beams. The beams should be divided into reasonable amounts of sub frames to accurately capture vertical earthquake effects.

Detailed information can be found in our publication:
Comparison of Practical Approaches for Modeling Shearwalls in Structural Analyses of Buildings

Comparison of Vertical Earthquake Analysis Results

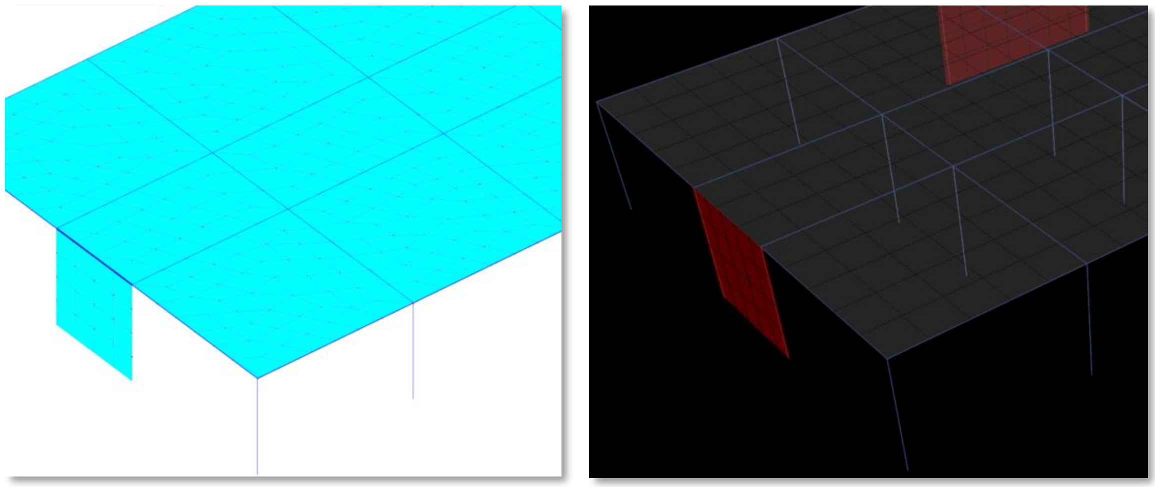
Axial Force values for **EZ** load case (Vertical EQ) for some selected columns and shearwall from Storey 1 in both software are given below.



EZ Load Case	1C14	1W4	1C16
ETABS	370.49	922.82	237.48
ProtaStructure	373.47	873.91	247.55

Note:
The results align well between the two software and differences are within reasonable limits. The differences are likely to be due to differences in mass calculation and mass distribution. Mesh size in slabs is also an affecting factor.

Default slab mesh is shown below for ProtaStructure and ETABS. Mesh frequency can be adjusted in both software. ProtaStructure also has an option to use **QUAD** elements in floor meshing.



Default floor mesh in ProtaStructure (on the left) and ETABS (on the right)

Conclusion and Discussion of Results

A comparison was made between two well-known and widely used structural engineering software, namely, ProtaStructure and ETABS. Export functionality from ProtaStructure to ETABS was not used and the model was created from scratch in both software. Horizontal and vertical acceleration spectra, storey mass and seismic weights, modal characteristics, base shear values and analysis results were compared.

The difference between the two software seems to be within reasonable limits. Although the difference in mass and structural weight calculation seems to be propagating to the other results, the difference is within expected tolerances.

ProtaStructure calculates the structural mass and weight in a more accurate manner, trying to avoid double counting and repetition. ProtaStructure also tends to use finer mesh by default in slab modeling to achieve greater load and mass decomposition resolution.

The model was kept simple for the sake of comparison. The following points may yield differences in the comparison of results, and they should be kept in mind in future studies:

- ProtaStructure can consider cracked/uncracked section in seismic/non-seismic combinations
- Two-stage analysis for buildings with basement floors can be carried out in ProtaStructure
- Differences may exist in the consideration of diaphragm eccentricity in EQS and RSA analyses between ProtaStructure and ETABS
- ProtaStructure automatic applies irregularity penalties, such as behavior factor adjustment, additional eccentricity amplification, etc.
- ProtaStructure automatically amplifies response spectrum analysis results to equivalent static
- Differences may exist in FE modeling approaches and member eccentricities between ProtaStructure and ETABS.
- ProtaStructure inserts members with accurate eccentricities, forming a detailed structural BIM model, while analytically transforming the columns to the centroidal locations. On the other hand, ETABS focuses on centerline modeling, usually simplifying or ignoring member eccentricities or locations.
- Difference in finite element member formulations may exist. There may be proprietary/different formulations for simulating membrane and plate bending, as well as beams.
- ProtaStructure automatic applies overstrength factors (where necessary) and live load reductions.

Important Note:

ProtaStructure is a highly advanced design software that performs detailed post-processing on analysis results according to code requirements. Especially for the seismic design, the raw analysis results are almost never directly used without modification. Hence, comparison studies to be done in the future should consider these factors and start from scratch by comparing simple results, gradually delving into complex output. The comparison is made to the best of the author's knowledge on ETABS. ETABS documentation should be referred to for detailed information or in case of any questions.

Thank You…

Thank you for choosing ProtaStructure Suite product family. Our top priority is to make your experience excellent with our software technology solutions.

Should you have any technical support requests or questions, please do not hesitate to contact us at all times through globalsupport@protasoftware.com and asiasupport@protasoftware.com

Our dedicated online support center and our responsive technical support team are available to help you get the most out of Prota's technology solutions.

The Prota Team